

***DESIGN APPLICATIONS USING
UNIGRAPHICS
STUDENT MANUAL
September 2002
MT10055 – Unigraphics NX***

Unigraphics Nx Student Manual

Unigraphics Solutions Inc

A red circular graphic with a gradient, appearing as a partial circle or a thick arc, located to the right of the company name.

Unigraphics Nx Student Manual:

The Hindu Index ,2003 **Practical Applications of Unigraphics** Unigraphics Solutions Inc,2000 Annual Index/abstracts of SAE Technical Papers ,2006 *Nx 11 Tutorial* Online Instructor,2016-10-08 NX 11 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 11 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations and simulation basics Table of Contents 1 Getting Started 2 Modeling Basics 3 Constructing Assembly 4 Generating Drawings 5 Sketching 6 Additional Modeling Tools 7 Top Down Assembly 8 Dimensions and Annotations 9 Simulation Hands on Tutorial Download Resource files from http://onlineinstructor.org/book/nx_11_tutorial **Nx 11 Tutorial** Tutorial Books,2016-10-05 NX 11 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 11 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations and simulation basics Table of Contents 1 Getting Started 2 Modeling Basics 3 Constructing Assembly 4 Generating Drawings 5 Sketching 6 Additional Modeling Tools 7 Top Down Assembly 8 Dimensions and Annotations 9 Simulation Hands on Tutorial Download Resource files from http://onlineinstructor.org/book/nx_11_tutorial **Nx 10 Tutorial** Online Instructor,2015-03-31 NX 10 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 10 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations and simulation basics Table of Contents 1 Getting Started 2 Modeling Basics 3 Constructing Assembly 4 Generating Drawings 5 Sketching 6 Additional Modeling Tools 7 Top Down Assembly 8 Dimensions and Annotations 9 Simulation Hands on Tutorial

NX 12 Tutorial Online Instructor,2020-11-16 NX 12 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 12 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations simulation basics product manufacturing information and rendering **Unigraphics NX for Total Product Engineering** Unigraphics Solutions

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 3D models You will learn to sketch on three different planes Front Top and Right planes You will use various sketching tools
 such as line rectangle circle and so on You will also learn to modify sketches using tools such as trim extend fillets and so on
 Learn to use geometric constraints and dimensions to achieve a definite shape and size of the sketch Sketches are converted
 into 3D features such as Extrude Revolve and so on You combine or subtract features to achieve the final part You can also
 add placed features sketch less features such as Fillets and Holes to the 3D geometry You explore mirroring and patterning
 commands to create repetitive features You will learn to use some additional modeling tools and work with multi body parts
 Learn to modify part geometry by editing sketches and feature parameters You explore Synchronous Modeling tools to
 modify the Part geometry by modifying its faces You build assemblies after creating parts There are two methods to build
 assemblies Bottom up and Top down In the Bottom up method you bring all the parts together and add constraints between
 them In the Top down method you create parts in the assembly level You explode assemblies to show the manner in which
 they were assembled You create Drawings of the parts and assemblies You insert part views and add dimensions and
 annotations to complete the drawing In case of assembly drawings you insert assembly views add Bill of Materials Balloons
 and Revision table The Sheet Metal design chapter covers various tools used to build sheet metal parts from scratch You will
 also learn to convert an existing part geometry into sheet metal part You also create flat patterns and 2D sheet metal
 drawings Finally you explore the surface modeling tools used to create complex shapes Table of Contents 1 Getting Started
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